

# Direct Numerical Simulations of turbulent flow in a driven cavity

R. Verstappen \*, J.G. Wissink, W. Cazemier, A.E.P. Veldman

*Department of Mathematics, University of Groningen, P.O. Box 800, 9700 AV Groningen, The Netherlands*

## Abstract

Direct numerical simulations (DNS) of 2 and 3D turbulent flows in a lid-driven cavity have been performed. DNS are numerical solutions of the unsteady (here: incompressible) Navier-Stokes equations that compute the evolution of *all* dynamically significant scales of motion. In view of the large computing resources needed for DNS cost-effective and accurate numerical methods are to be selected. Here, various-order accurate spatial discretization methods for DNS have been evaluated by applying them to the 2D driven cavity at  $Re = 22,000$ . To analyze the results of the DNS of the 2D flow in a driven cavity at  $Re = 22,000$  the proper orthogonal decomposition (POD) technique has been applied. POD is an unbiased method to determine coherent structures. The Galerkin projection of the Navier-Stokes equations on the space spanned by the POD-basis-functions yields a relatively low-dimensional set of ordinary differential equations that mimics the dynamics of the Navier-Stokes equations. 3D DNS with no-slip conditions at all walls of the cavity have been performed at both  $Re = 3,200$  and  $Re = 10,000$ . The results reproduce the experimentally observed Taylor-Görtler-like vortices.

**Key words:** Direct Numerical Simulation; Turbulence; Navier-Stokes equations; Driven cavity; Numerical methods; Proper orthogonal decomposition

## 1. Introduction

Direct Numerical Simulations (DNS) are numerical solutions of the unsteady (here: incompressible) Navier-Stokes equations that compute the evolution of *all* dynamically significant scales of motion. Unfortunately, DNS of engineering flows exhaust the largest available computing resources by requiring machines in the exa( $10^{18}$ ) flops range with exabytes of memory. Thus, for engineering applications acceptable computational effort can only be obtained by modelling

the turbulence of the flow. However, with existing turbulence models the simulation accuracy required by industry cannot be reached in many technological applications, making turbulence modelling the main pacing item in the development of applied CFD codes. It is generally expected that DNS will play the key role in obtaining better turbulence models; DNS have now become the chief source of highly detailed data for turbulence modelers and researchers [1].

In this paper, we report on DNS of turbulent flows in two- and three-dimensional, lid-driven, cavities and on their mathematical analysis. The two-dimensional simulations have been performed to evaluate numerical methods for DNS

\* Corresponding author.

and to study the properties of the proper orthogonal decomposition (POD) technique.

## 2. DNS of two-dimensional flow in a driven cavity

In two dimensions, the flow converges to a steady state for Reynolds numbers up to 10,000 [2]. For larger Reynolds numbers, 2D driven cavity flows undergo a transition to unsteadiness. Our direct simulations exhibit that the flow converges to a purely periodically oscillating state at  $Re = 11,000$ . The amplitude of the periodic motion is small: in terms of the total kinetic energy 0.2% of the mean value (more details can be found in [3]). DNS of the motion of the vortical structures at  $Re = 22,000$  reveals that the dynamics has become chaotic: the correlation dimension is approximately 2.8, the Kolmogorov entropy is approximately 3.0. As expected, the number of vortices has increased compared to the simulation at  $Re = 11,000$ . Furthermore, the motion of vortical structures is no longer confined to small regions near the corners of the cavity. To illustrate this, three instantaneous vorticity fields are shown in Fig. 1.

## 3. Evaluation of numerical methods for DNS

In view of the large computing resources needed for DNS a careful selection of cost-effec-

tive and accurate numerical methods is important. Here, various-order accurate spatial discretization methods for DNS have been evaluated by applying them to the driven cavity at  $Re = 22,000$ . The evaluation focusses on the accuracy with which the mean kinetic energy is computed using a  $200^2$  economically stretched, staggered grid, and a time-step that is so small that the error resulting from the time-integration is negligible. Central as well as upwind discretizations have been considered. The central method consisting of a  $m$ th order central discretization of the convective term (in conservative form) combined with a  $n$ th order interpolation, and a  $k$ th order central discretization of the diffusive term is denoted by  $CmInCk$ . Likewise, an upwind method is denoted by  $UmInCk$  if the conservative term (now in non-conservative form) is discretized using a  $m$ th order upwind-biased method. A Lagrange interpolation is used to obtain the coefficients of the stencils. Since stencils with more than one gridpoint on either side of the point at which the derivative is to be calculated cannot be applied near the boundaries, high-order methods are gradually replaced by lower-order methods in the vicinity of the boundaries. Reference data for the comparison is obtained using the  $U514C6$  method on a  $400^2$  stretched grid. The main results of the comparison are shown in Fig. 2. From this figure, it can be concluded that  $C414C6$  and  $U716C8$  perform the best of the methods considered here.

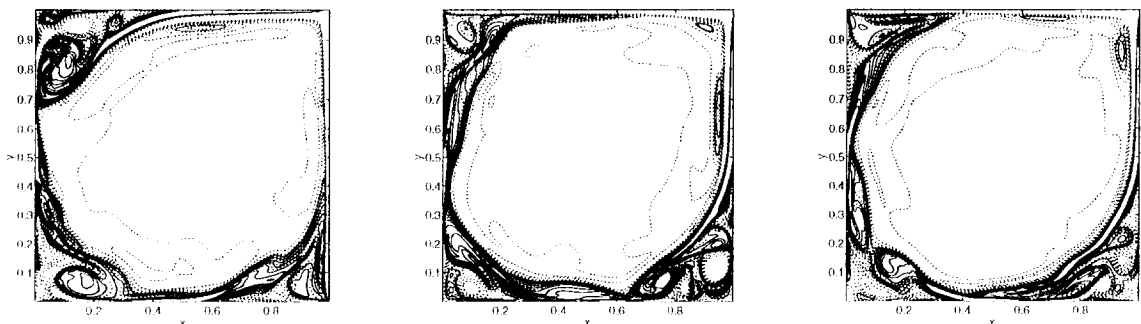


Fig. 1. Snapshots of the vorticity in a 2D driven cavity ( $Re = 22,000$ ).

#### 4. POD analysis of DNS

To analyze the results of the DNS of the turbulent flow in a driven cavity at  $Re = 22,000$ , we make use of proper orthogonal decomposition (POD). That is (see e.g. [4]), from fluctuating flow fields a set of (say  $N$ ) basis-functions is extracted, which is optimal in the sense that any other decomposition (e.g. in  $N$  Fourier-modes) cap-

tures less kinetic energy. The Galerkin projection of the Navier-Stokes equations on the space spanned by the basis-functions yields a relatively low-dimensional set of ordinary differential equations. This ‘cheap’ (compared to Navier-Stokes) set can be analyzed using tools from dynamical systems theory, and thus we can advance our understanding of turbulence, provided that this set mimics the dynamics of the Navier-Stokes

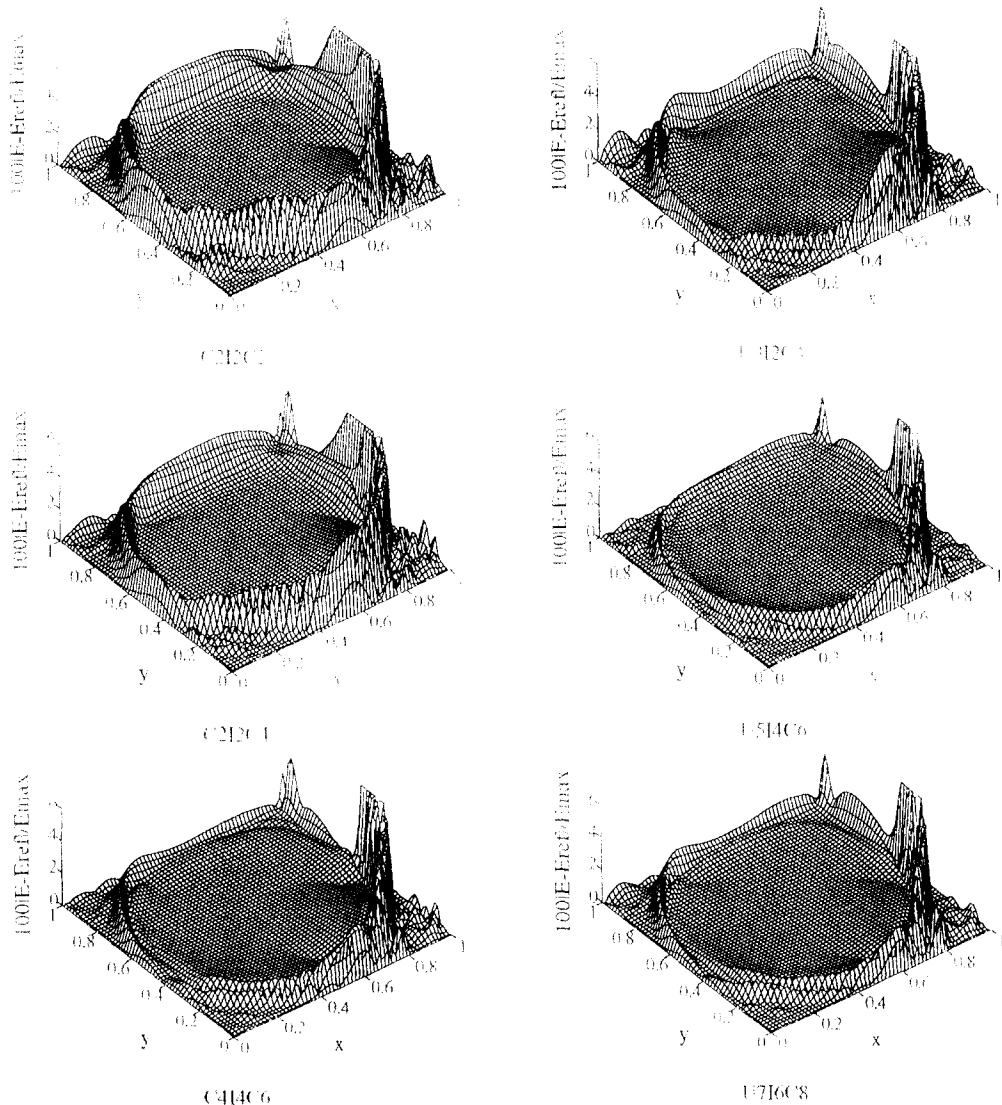


Fig. 2. Errors in the mean kinetic energy for a number of spatial discretization methods (2D driven cavity,  $Re = 22,000$ ).

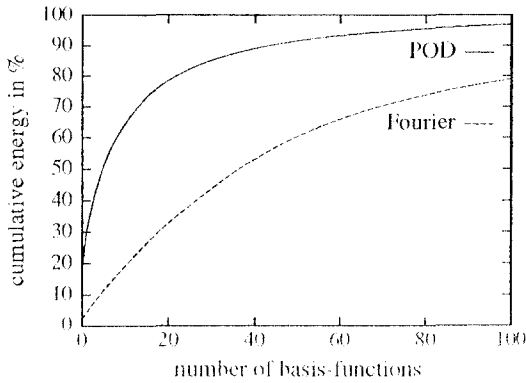


Fig. 3. Comparison of POD and Fourier-modes.

equations. To investigate how well it does this, we have performed a direct simulation of the turbulent flow in a 2D driven cavity at  $Re = 22,000$  during more than 700 large-eddy turn-around times to obtain the data necessary to compute up to 700 basis-functions. These basis-functions are computed using the so-called snapshot method [5]. The energy captured by POD-modes has been

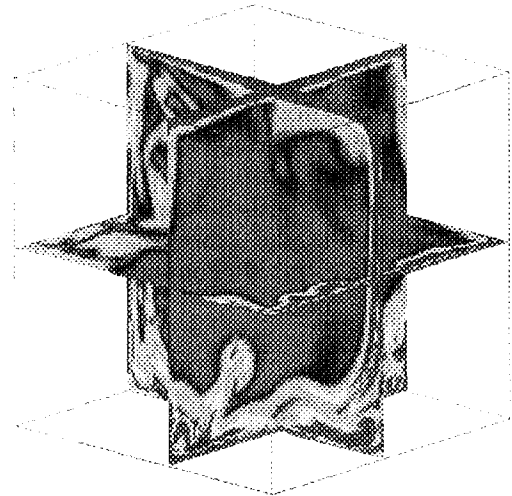


Fig. 5. Snapshot of the vorticity in a 3D driven cavity at  $Re = 10,000$ .

compared to the energy captured by Fourier-modes (Fig. 3), and, as expected, POD is far superior to Fourier. Furthermore, as can be seen

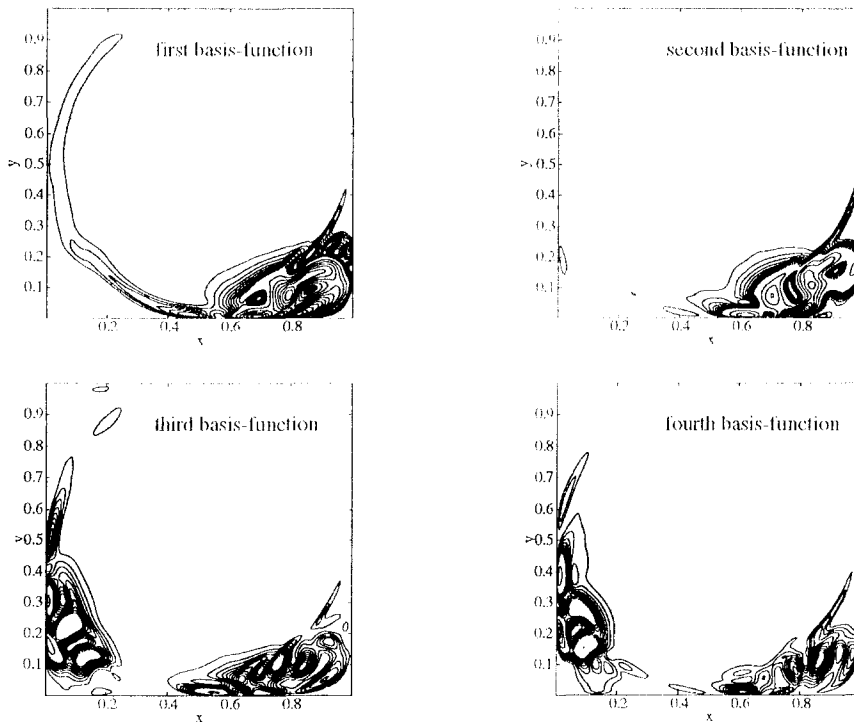


Fig. 4. The first four POD basis-functions.

in Fig. 4, POD is an unbiased method to determine the coherent structures (eigenflows) in the driven cavity flow.

It has been found that the evaluation obtained by numerical integration of a 80-dimensional dynamical system, which is obtained by projecting the Navier-Stokes equations on the first 80 POD basis-functions, agrees well with direct numerical simulation data at  $Re = 22,000$ . Also, when the Reynolds number in the 80-dimensional dynamical system is taken equal to 11,000 the integration of the dynamical system reproduces the periodic solution as obtained by DNS at  $Re = 11,000$ .

### 5. DNS of three-dimensional turbulent flow in a driven cavity

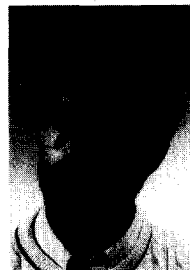
Reliable experimental results (see e.g. [6]) have pointed out that the flow in a three-dimensional driven cavity is already unsteady for Reynolds numbers much smaller than 10,000. The experiments reveal that the flow becomes turbulent at Reynolds numbers between 6,000 to 8,000, and that the first instability in the 3D flow is certainly not two-dimensional: the experimentally observed instability is associated with 3D Taylor-Görtler-like vortices. Thus, fluid flows in the direction perpendicular to the lid-motion are significant. For this reason, full 3D DNS (with no-slip conditions at all walls of the cavity) have been performed at  $Re = 3,200$  and  $Re = 10,000$ . A  $100^3$  economically stretched, staggered grid has been used to solve the incompressible Navier-Stokes equations numerically. The CPU-time on one processor of NLR's NEC SX-3 is  $0.6 \mu s$  per gridpoint and time-step. The results reproduce the experimentally observed Taylor-Görtler-like vortices, and the computed mean velocities agree well with those measured by Koseff and Street [6]. As an illustration of the structures in the flow, Fig. 5 shows an instantaneous vorticity field in the 3D cavity at  $Re = 10,000$ . In this figure, the orientation of the cavity is such that the upper-plane is driven from the left/back to the right/front.

### Acknowledgements

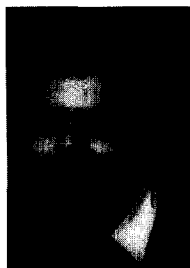
Both the Stichting Nationale Computerfaciliteiten (National Computing Facilities Foundation, NCF) with financial support from the Nederlandse Organisatie voor Wetenschappelijk Onderzoek (Netherlands Organization for Scientific Research, NWO) and the National Aerospace Laboratory NLR are gratefully acknowledged for the use of supercomputer facilities.

### References

- [1] W.C. Reynolds, in: *Wither Turbulence? Turbulence at the crossroads*, J.L. Lumley (ed.) (Springer-Verlag, Berlin, 1990), 313.
- [2] U. Ghia, K.N. Ghia and C.T. Shin, *J. Comp. Phys.* 48 (1982) 387.
- [3] R. Verstappen, J.G. Wissink and A.E.P. Veldman, *Appl. Sci. Research* 51 (1993) 377.
- [4] N. Aubry, P. Holmes, J.L. Lumley and E. Stone, *J. Fluid Mech.* 192 (1988) 115.
- [5] L. Sirovich and M. Kirby, *Physica D* 37 (1987) 126.
- [6] J.R. Koseff and R.L. Street, *ASME J. Fluids. Eng.* 106 (1984) 21.



Roel Verstappen was born in Venray (The Netherlands) on 22 June 1962. He received his education in applied mathematics at the University of Twente in the Netherlands. In 1989 he received his Ph.D. in engineering from this university. After that he moved to the University of Groningen where he works on the field of computational fluid dynamics.



After a working period of several years as an analyst at Delft Hydraulics, Jan Wissink started to study mathematics at the University of Groningen. Since his graduation he is a Ph.D. student, making researches into numerical methods (especially finite-volume methods) for direct numerical simulation of turbulence.



Willem Cazemier studied mathematics at the University of Groningen. The last year of his study he spent at the National Aerospace Laboratory (NLR) in Amsterdam. After his graduation in 1992, he became a Ph.D. student in Groningen, where he is doing research on the Proper Orthogonal Decomposition to study turbulence.



Arthur E.P. Veldman obtained a Ph.D. in Applied Mathematics from the University of Groningen. In 1977 he joined the National Aerospace Laboratory NLR in Amsterdam, where he was involved in various projects in the area of computational aero- and hydrodynamics. In addition, between 1984 and 1990 he was part-time professor of CFD at Delft University of Technology. In 1990 he returned to Groningen, where he now occupies the chair in Computational Mechanics.